3.6 User Defined Inelastic Material Model

MAC/GMC, has the option for a user to implement their own inelastic constitutive model. This is accomplished by using the subroutine **USRMAT** into which the user writes the necessary FORTRAN code for the particular constitutive model being implemented. The **USRMAT** subroutine is as shown below and is always called whenever ncmd=99 (see section **4.2.12**).

```
c#######################################################################
       SUBROUTINE USRMAT(DSA, SA, PE, PV, D, LOCTISO, TIME, TSTEP,
     &     CTEMP, DTEMPR, NIO, NE, NV, NS, MN, CDUM, DMGF,
     &     NEP, NVP, NSASIZE)

c     purpose:  user material constitutive model for determination of
c               the inelastic strain and state variable rates
c               (used when ncmd = 99)

      IMPLICIT DOUBLE PRECISION (A - H, O - Z)


      CHARACTER*2 CDUM
      DIMENSION SS(6), S(6), R(6)
      DIMENSION DSA(NSASIZE), SA(NSASIZE)
      DIMENSION PV(NVP), PE(NEP), D(3)

**********************************************************************
c  note: 1) in this subroutine, [SA] and [DSA] contain the
c           micro (subcell) quantities for aboudi’s micromechanics model
c
c        2) arrangement of [dsa] & [sa]  arrays:
c           variable             location
c         +-------------------------------------------------------------
c         | strain rate          (1-6)  (contains ENGINEERING shears)
c         |-------------------------------------------------------------
c         | stress rate          (7-12)
c         |-------------------------------------------------------------
c         | inelastic
c         | strain rate          (13-18)  (contains ENGINEERING shears)
c         |-------------------------------------------------------------
c         | 12 “slots”           (19-30)
c         | for state variables
c         |-------------------------------------------------------------
c         | thermal strain rate  (31-36)
c         +-------------------------------------------------------------
***********************************************************************
c NOTE:  quantities in [SA] and [DSA] are SUBCELL quantities - the
c        values on entry are for the first subcell containing material
c        # MN - the values on exit of this subroutine will be applied to
c        ALL SUBCELLS containing material # MN.  It is thus recommended
c        that, if using the field variables, you assign the appropriate
c        material # to ONE SUBCELL ONLY.  Use of [SA] and [DSA] in this
c        context in conjunction with bending in laminate theory will
c        result in erroneous results as field variables become dependent
```

```
c        on through-thickness position.
*************************************************************************
c   on entry:
c        SA        - vector of total (integrated) quantities (see above)
c        PE(NEP)    - vector of elastic constants for material MN
c                     (where NE = # of elastic constants --> 9 MAX)
c        PV(NVP)    - vector of viscoplastic constants for material MN
c                    (where NV = # of viscoplastic constants --> 19 MAX)
c        D(3)      - vector of direction cosines (for models 3, 7, & 9)
c        LOCTISO   - flag indicating if ANY material exhibits
c                    local transverse isotropy (and global anisotropy)
c                  = 0 - all materials are at most globally transversely
c                        isotropic (D not used)
c                  = 1 - at least one material is locally transversely
c                        isotropic (D used)
c        TIME      - current time
c        TSTEP     - current time step
c        CTEMP     - current temperature
c        DTEMPR    - time rate of change of temperature
c        NIO       - unit number of .out file
c        NE        - # of elastic constants --> 9 MAX
c        NV        - # of viscoplastic constants --> 19 MAX
c        NS        - subcell number
c        MN        - material number
c        CDUM      - material character/number designation
c                    (i.e. F1 = fiber #1)
c        DMGF      - damage factor - if damage is included the user
c                    should multiply material stiffness terms by DMGF
c                    when using such terms in their inelastic model.
c
c   expected on exit:
c        DSA       - vector of rate quantities (see above)
*************************************************************************
c
*************************************************************************
*                      BEGIN USER EDITS                                 *
*************************************************************************
c        place code defining model here
*************************************************************************
*                        END USER EDITS                                 *
*************************************************************************
      RETURN
      END
```

A description of the input and output required for the usrmat subroutine is as follows

Data supplied to **USRMAT**:

SA	array containing current total quantities for all of the state variables
PE	array containing elastic constants

PV	array containing inelastic constants
D(3)	vector of direction cosines (for models 3, 7, & 9)
LOCTISO	flag indicating if ANY material exhibits local transverse isotropy (and global anisotropy) = 0 - all materials are at most globally transversely isotropic (D not used) = 1 - at least one material is locally transversely isotropic (D used)
TIME	current time
TSTEP	current time increment (step)
CTEMP	current temperature
DTEMPR	current temperature rate
NE	total number of elastic constants
NV	total number of inelastic constants
NIO	output file unit number
NCE	current subcell number
NS	subcell number
MN	material number of current constituent
CDUM	Character string identifying current constituent
DMGF	damage factor - if damage is included the user should multiply material stiffness terms by DMGF when using such terms in their inelastic model.

Output expected from **USRMAT**:

DSA	current increments in all state variables

The state variables are arranged in SA and DSA in the following order:

Position	Quantity
1-6	strain
7-12	stress
13-18	inelastic strain
19-30	available space for 2 6x1 vectors for model dependent internal state variables
31-36	thermal strain
37-42	debonding parameters

Again, SA contains the total quantities and DSA contains the rates.

☞ **Note: Example K** contains a sample input file and the **USRMAT** subroutine containing an implementation of the Bodner-Partom viscoplastic material model as well as two elastic models.

☞ **Note: GMC** utilizes **engineering strains,** in SA and DSA consequently it is the users responsibility to convert tensorial strain quantities to engineering **before exiting** the **USRMAT** routine.